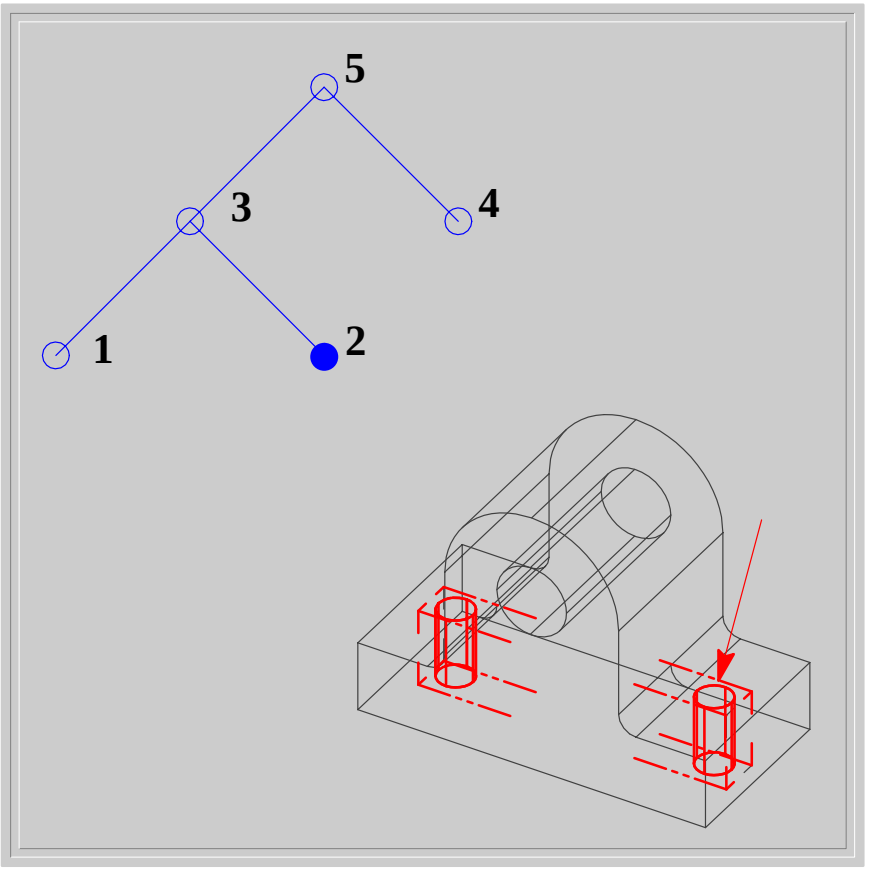


Using History Access to Modify Features

I-DEAS® Tutorials: Fundamental Skills

Learn how to:

- select nodes on the history tree
- add and delete edges to fillet features
- insert a feature using *History Access*
- suppress features

Before you begin...

Prerequisite tutorials:

1. Getting Started (I-DEAS® Multimedia Training)

   –or–

   Introducing the I-DEAS Interface,
   Quick Tips to Using I-DEAS
   –and–
   Creating Parts

2. Sketching and Constraining
3. Dimensioning
4. Building Sections
5. Using Sketch Planes and Understanding Sketch Pads
6. Using Sections and Sketch Planes
7. Extruding and Revolving Features
8. Adding Features with Associativity
9. Modifying Features

Open the model file that you created from the previous tutorial, Modifying Features.

File
Open

Open Model File form

Model File name: modify
OK

Make sure you're in the following application and task:

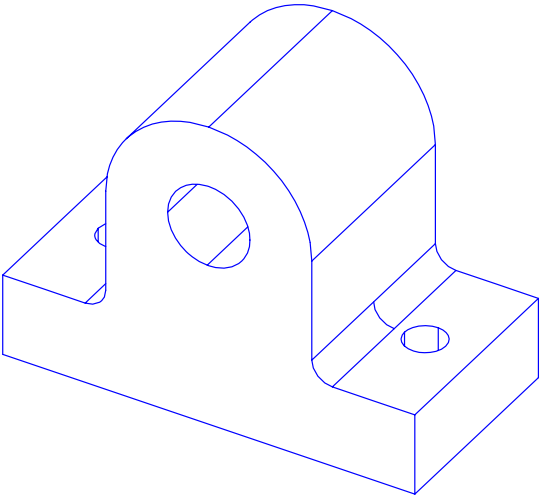
Design, *Manufacturing*, or *Simulation*
Modeler

Verify your units are set to mm.

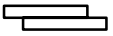
Options
Units

mm (*milli newton*)

Result



Save your model file.


File
Save

Warning!
If you are prompted by I-DEAS to save your model file, respond:


No

Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

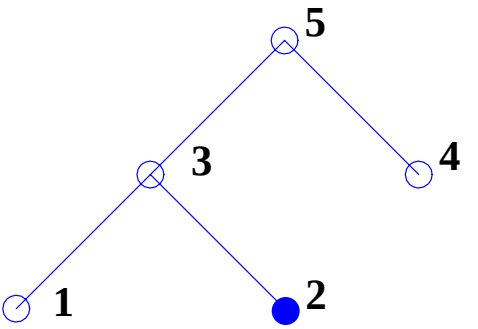
Hint
To reopen your model file to the previous save, press Control-z.

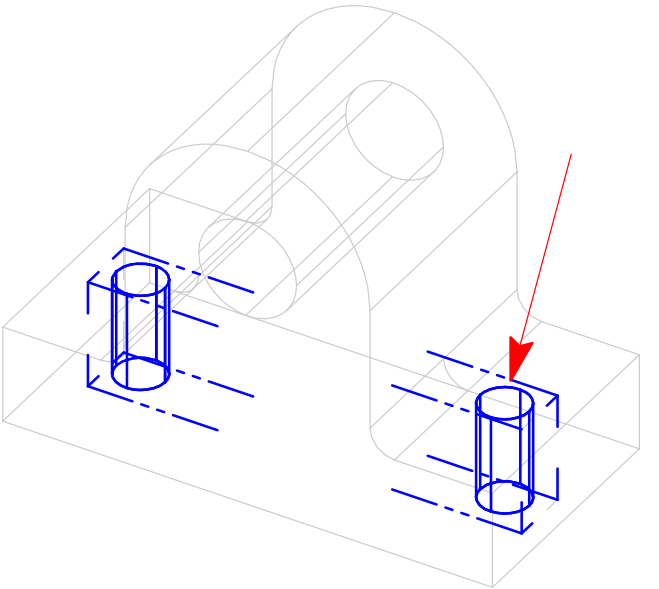
Select nodes on the history tree 1 of 2

You can modify features by picking the history tree of the part. A history tree represents the entire process for creating a part.

The history tree is composed of nodes. The nodes indicate the operations used at each stage in the part's creation. You can select these nodes to modify or insert a feature, or to access the underlying wireframe.

In the next step, you'll select the nodes composing the history tree for your part.





Select nodes on the history tree 2 of 2

To select features by history access, pick either the *History Access* icon or the right mouse button menu option.

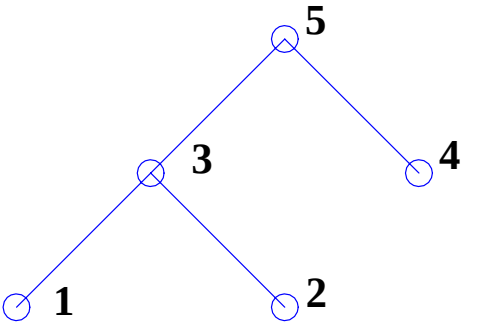
Practice selecting the features on the history tree.

–or–

 *History Access*...

 pick anywhere on the part

1. extrusion
2. extrusion
3. cut
4. fillet round info
5. fillet round



Things to notice

Notice how the surfaces of the selected feature are highlighted and the feature is boxed in dashed yellow-shaded lines.

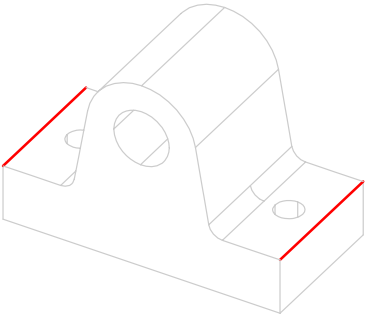
History Access form

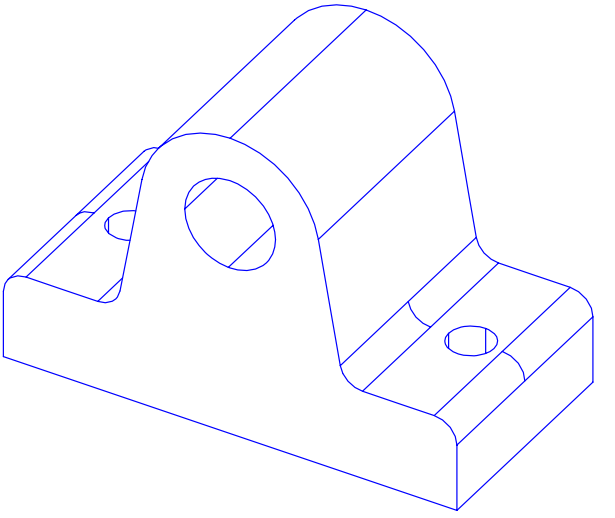
 *Dismiss*

Add and delete edges to fillet features 1 of 3

When you select a fillet feature, you can still add or delete edges, as well as change dimensions. In the next steps, you'll access the history tree and add fillets.

By modifying the existing fillet feature rather than adding a new feature, you keep fewer steps in the history tree.





Add and delete edges to fillet features 2 of 3

Select the fillet feature and add two new edges to it.

 pick anywhere on the part

 (Accept)

History Access form

 pick the fillet round info node (upper right)

Things to notice

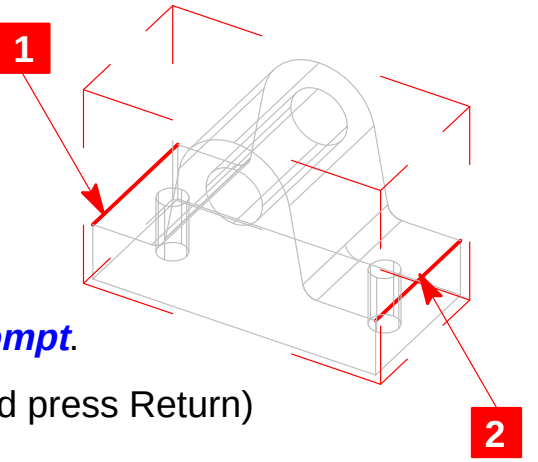
Notice that *Feature Parameters* is not one of the options for this type of feature.

 *Add/Change*

1

2 shift-pick

 (Done)

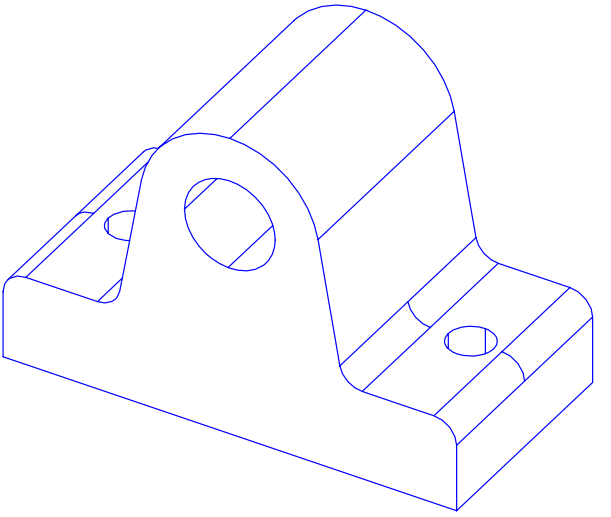


 **Check *I-DEAS Prompt*.**

 5 (type in and press Return)

 *Done*

Add and delete edges to fillet features 3 of 3



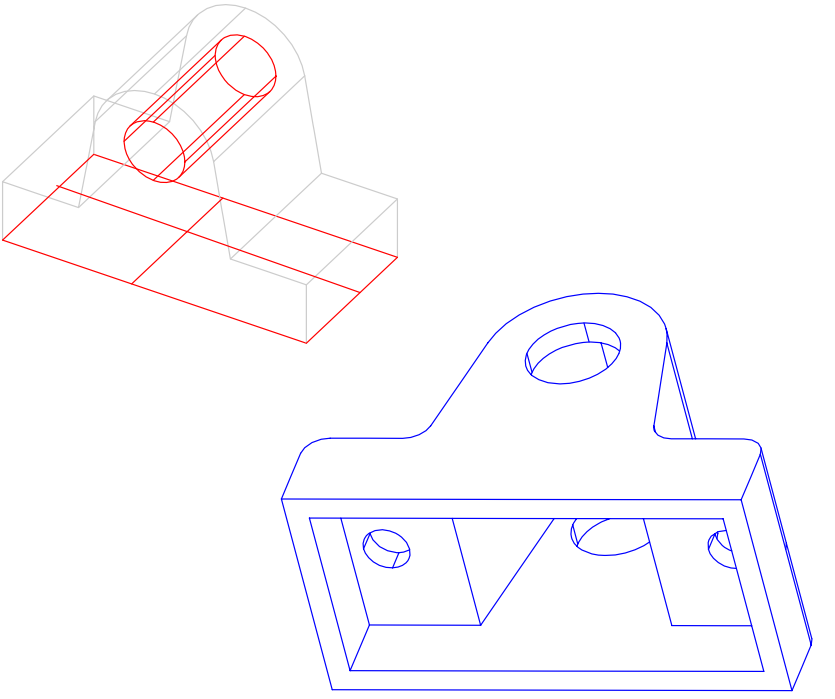
Recovery Point

File
Save

Insert a feature using *History Access* 1 of 4

Sometimes the order of features is important. To insert a feature in the middle of the history tree, the part can be rolled back to that point in the history tree, the new feature added, and the part updated.

As an example of this, in this section, you will insert a shell feature before the hole cutout and the fillet.



Insert a feature using *History Access* 2 of 4

Select the first feature using *History Access* and roll back to that point.

pick anywhere on the part

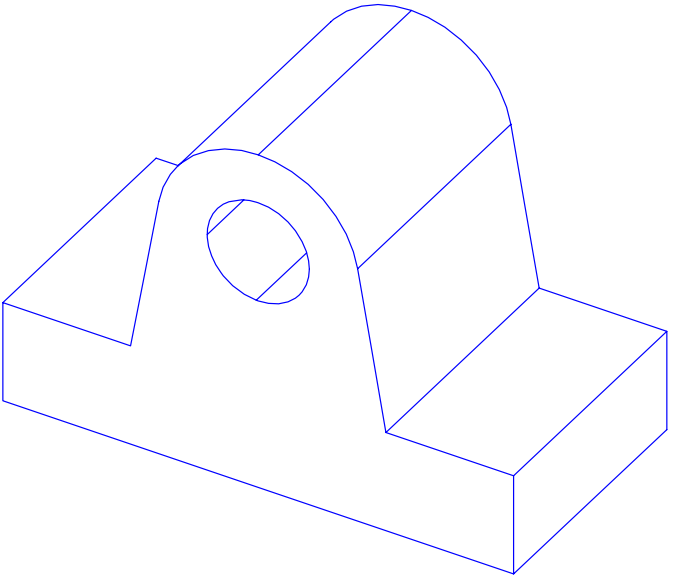
(Accept)

History Access form

pick first feature on history tree (bottom left)

Rollback

Dismiss

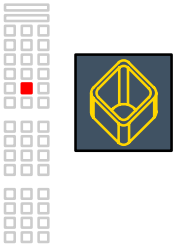


Things to notice

This is the part rolled back to where it was after you created the first feature.

Insert a feature using *History Access* 3 of 4

Shell the part with a thickness of 5mm, deleting the bottom face and the inner cylinder face.



pick anywhere on part

(Done)

Shell form

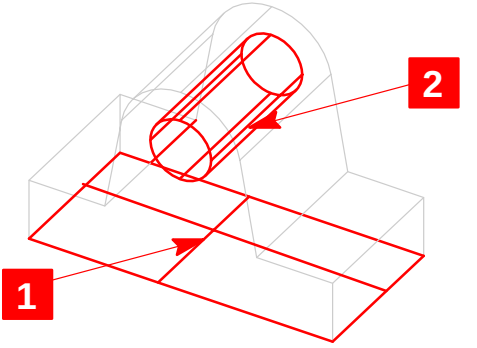
Thickness: 5

---

1 pick bottom face

2 shift-pick cylinder

(Done)

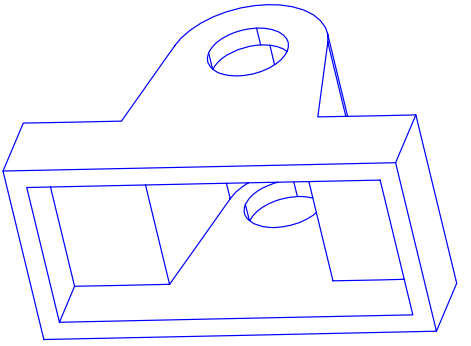


Shell form

OK

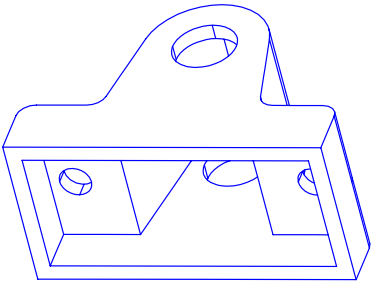
If you created your part with different dimensions and get an error on the *Shell* command, you may need to use a lower value for the thickness, or shell to the outer direction instead of inward, using:

Insert a feature using *History Access* 4 of 4



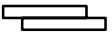
Update the part.

If you get a message indicating that you are in a rolled back state, select *OK* to continue.



To have shelled the part as the last feature without rolling back would not have had the same result.

Recovery Point



File
Save

Suppress features 1 of 3

You can suppress features. This leaves their definition in the history tree, but turns them off so they can’t be seen on the resulting part.

Select the shell info leaf on the history tree and suppress the feature.

 pick anywhere on part

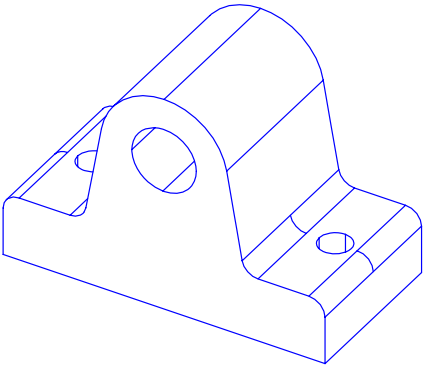
 (Accept)

History Access form

 pick shell info feature on history tree

pick icon on form

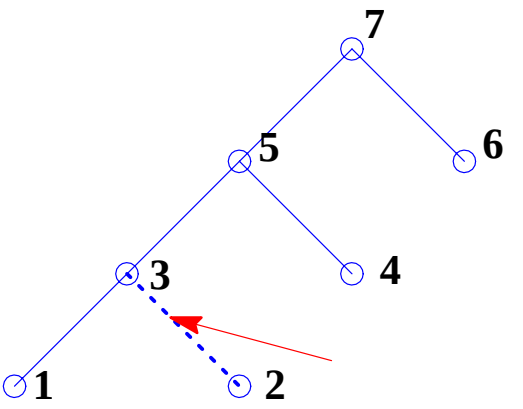
 *Dismiss*



Things to notice

Now it is a solid part without the shell feature. More information on shell features is given in the tutorial “Adding Fillet, Shell, and Draft Features.”

Display the history tree again.



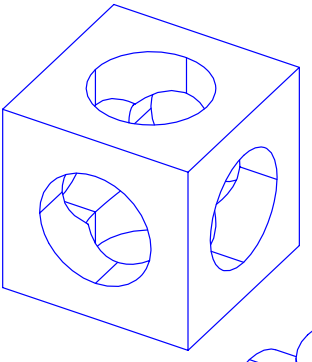
1. extrusion
2. shell info
3. shell
4. extrusion
5. cut
6. fillet round info
7. fillet round

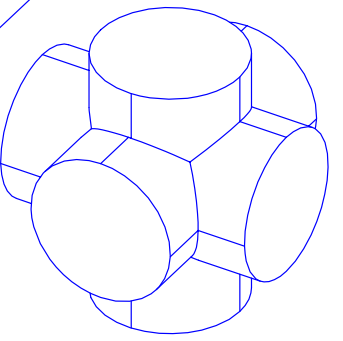
Things to notice

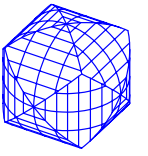
Notice that the shell feature is shown as a dashed line, indicating that it is suppressed.

Before quitting the tutorial, try the “On your own” exercise on the next few pages. If you’d rather try the “On your own” at a later time, skip to the last page for wrap-up instructions.

What do these three parts have in common?



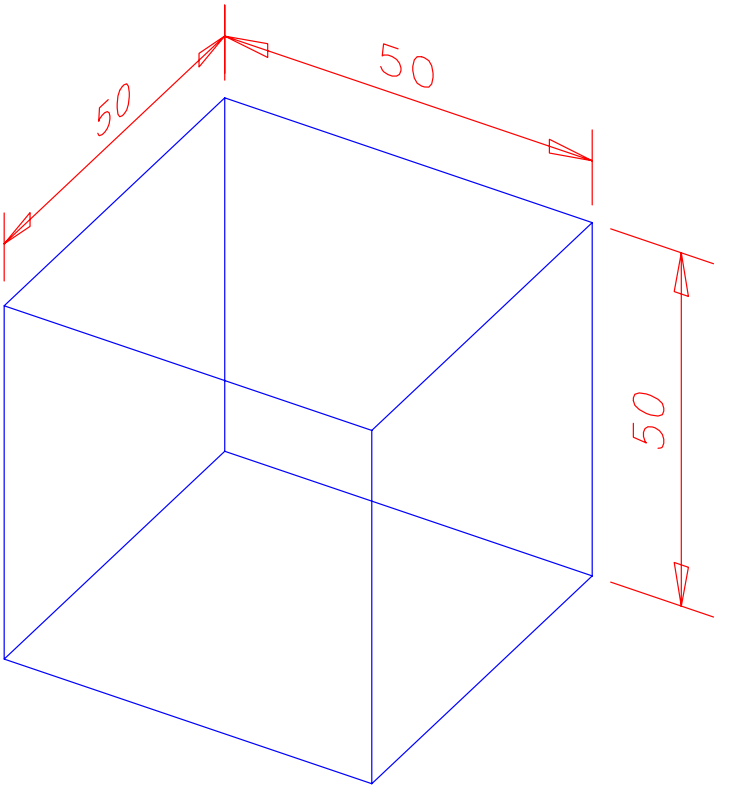




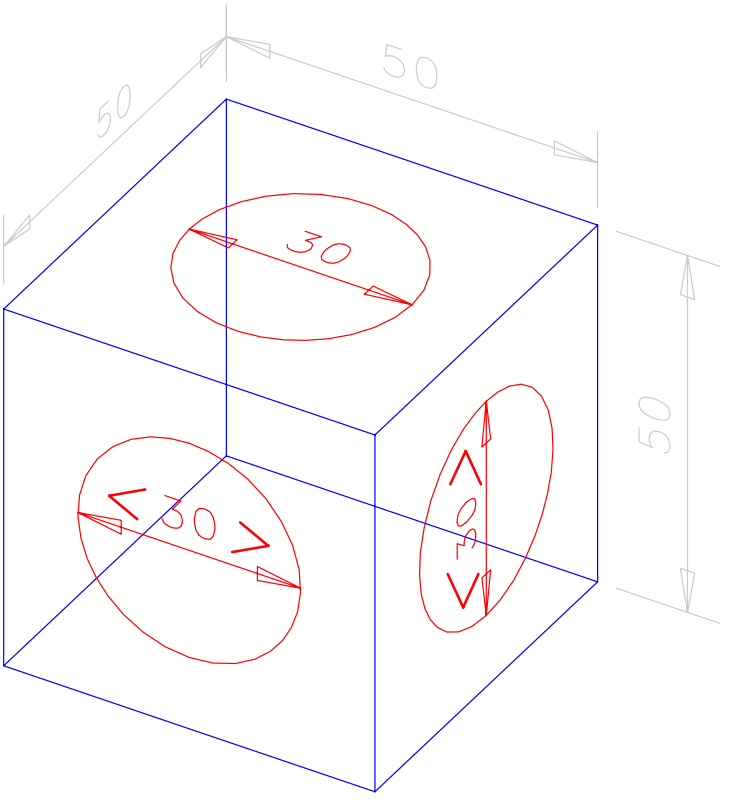
Actually, it's the same part. To change the part, you only have to modify one of its parameters for a feature.

Try this on your own. If you need help, refer to the next few pages, which give you hints on how to create the three parts.

Step 1. Build the part.



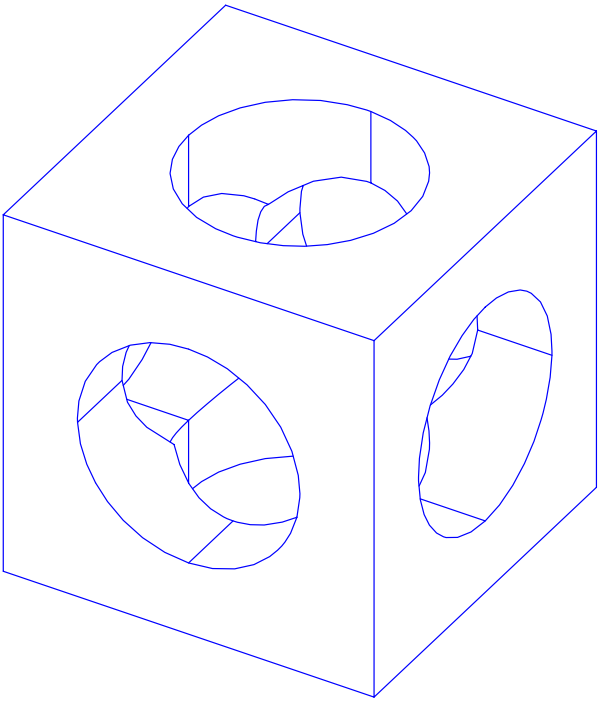
Step 2. Sketch a circle centered on each face, with the same diameter for each.



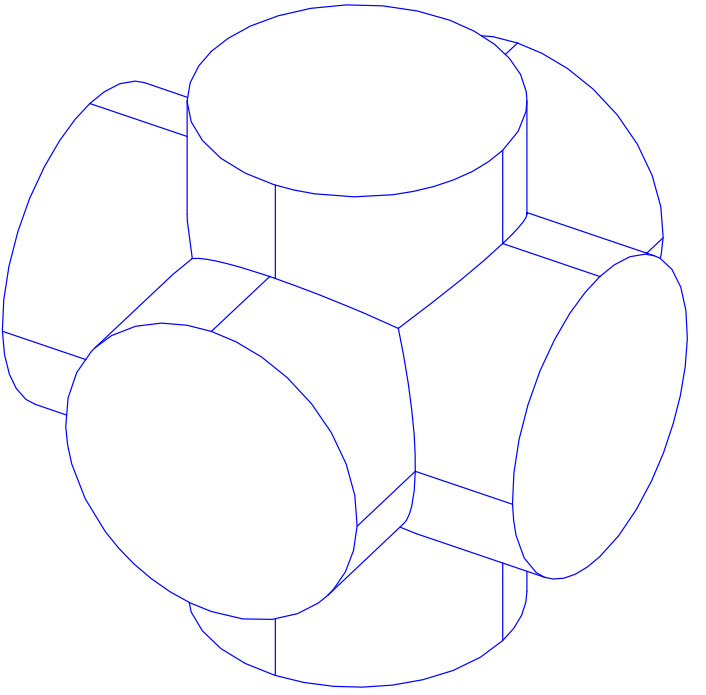
Hint

Use *Sketch in Place* before sketching each circle.

Step 3. Cut out each hole using *Thru All*.



Step 4. Using the history tree, modify the feature parameters of each of the last three extrude features to be a protrude instead of a cutout. Then, suppress the original block.



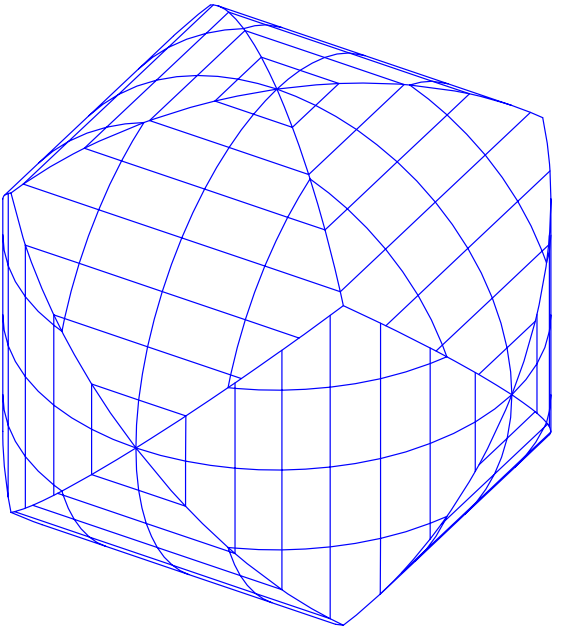
Check *I-DEAS List*.

A warning indicates the sketch planes are out-of-date for the wireframe. This is because you suppressed the associated surfaces. Pick *Continue Replay* to update the part for the three protrusions.

Replay Interrupted form

Continue Replay

Step 5. Next, modify the last three features to be *Intersects* and unsuppress the original block.



Tutorial wrap-up

You have completed the Using History Access to Modify Features tutorial.

Delete or put away the parts. These parts are not used in any other tutorials.